

TIME DOMAIN SIMULATION OF AIRFOIL FLUTTER USING FLUID STRUCTURE COUPLING THROUGH FEM BASED CFD SOLVER

Davinder Rana⁺, Sandeep Patel⁺, Amit Kumar Onkar^{*} and M. Manjuprasad[#]
National Aerospace Laboratories-CSIR, Bangalore, India

ABSTRACT

The objective of this work is to study the flutter characteristics of an airfoil in a two-dimensional subsonic flow by externally coupling a Reynolds Average Navier Stokes (RANS) based CFD solver with the in-house structural code in time domain. The ANSYS FLOTRAN CFD, a finite element based computational fluid dynamics solver, is adopted here to generate the aerodynamic pressure distributions on an airfoil section in subsonic regime. The airfoil dynamics is accordingly simulated through external coupling of the ANSYS FLOTRAN CFD solver with a 2DOF airfoil structural model through a Newmarks's implicit time integration scheme. A symmetric NACA 0012 airfoil of unit chord is chosen for the analysis, with suitable spring stiffness and inertia values so that flutter instability occurs in the subsonic regime. Unsteady motion in the fluid-structure system is numerically simulated with small initial conditions. In the present analysis, the airfoil is not allowed to move and the pitch angle of airfoil is assigned to the air flow at the inlet boundary of the domain. Flutter boundary is indicated by the critical free stream flow velocity (and dynamic pressure) beyond which oscillation amplitudes diverge in time.

Key Words: Time domain analysis, airfoil flutter, Newmarks's algorithm, ANSYS FLOTRAN CFD solver

NOMENCLATURE

c	Chord length of airfoil
h	Heave (positive upward) displacement
α	Pitch (positive nose up) displacement
m	Mass of airfoil section
M_∞	Free stream Mach number
C_h	Structural damping in heave motion
C_α	Structural damping in pitch motion
K_h	Spring stiffness in heave motion
K_α	Spring stiffness in pitch motion
ζ_h	Heave damping ratio
ζ_α	Pitch damping ratio
I_α	Mass moment of inertia about support point
X_{cp}	Position of centre of pressure of airfoil
X_{cm}	Position of centre of mass of airfoil
X_o	Position of centre of flexural axis of airfoil

⁺Project staff, Structural Technologies Division, NAL

^{*}Scientist, Structural Technologies Division, NAL

[#]Senior Principal Scientist, Structural Technologies Division, NAL

1. INTRODUCTION

The understanding of aerodynamic flows and their interactions with structures is becoming increasingly important for aerospace vehicles. Since airplane structures are not completely rigid, the aeroelastic phenomena arise due to structural deformations induced by aerodynamic forces. The simulation of the aeroelastic phenomena requires an integrated analysis of fluids and structures. Closed-form solutions are available for aeroelastic computations when flows are either in the linear subsonic or supersonic range. Excellent treatises on classical aeroelasticity have been presented in [1-3]. In these texts, the dynamics of the airfoil under quasi-steady aerodynamic flow has been discussed in detail. Manjuprasad *et al* [4] and Onkar *et al* [5] also developed a suitable algorithm for simulating flutter of an airfoil in the time domain using FEM based inviscid and laminar viscous CFD solver respectively. However, in general the flow around the body is turbulent in nature. For accurate lift and moment computation, the Navier-Stokes CFD solver along with an appropriate turbulent model should be used. Hence, to obtain an accurate aeroelastic response, it is necessary to solve the Navier-Stokes equations incorporating suitable turbulent model and couple them with the structural equations.

This paper presents a procedure for solving fluid-structure interaction problems of airfoil in two-dimensional subsonic flow [6]. The solution of fluid flow problems are based on the Navier-Stokes equations. The standard k- ϵ turbulent model is employed for the solution of two-dimensional viscous flow problems on unstructured meshes. The

computational capability of the FLOTRAN CFD solver is investigated by solving flow over NACA 0012 airfoil under steady flow condition. The pressure distributions, their integrated values of lift coefficients and the centre of pressure for a symmetric NACA 0012 airfoil are determined for various angles of attack under steady flows. Then, an external code in FORTAN is developed to accurately couple the FLOTRAN CFD solver with the structural equations in the time domain using Newmark's algorithms. Results are demonstrated for aeroelastic responses of airfoil using the coupled fluid-structure code in subsonic flow regimes. The airfoil stiffness characteristics are chosen so that it has a subsonic flutter boundary. Flutter frequencies and the corresponding flutter velocities obtained by the various methods are also compared.

2. PROBLEM FORMULATION

The fundamental governing equations of fluid dynamics are continuity, momentum and energy equations. The continuity equation in differential form is given as [7]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (1)$$

where, ρ is the density, and u , v and w are the components of velocity in the x , y and z directions.

For compressible solution algorithm, the derivative of density with respect to pressure is derived from the ideal gas equation *i.e.* $p = \rho R T$. Hence, the above equation can be expressed as:

$$\frac{1}{RT} \frac{\partial p}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (2)$$

The momentum equation in x direction derived from the Newton's second law of motion as:

$$\begin{aligned} \frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + R_x + \\ \mu_e \left[\frac{\partial}{\partial x} \left(\frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left(\frac{\partial u}{\partial z} \right) \right] + T_x + \rho f_x \end{aligned} \quad (3)$$

Similarly, the momentum equation in y and z directions can be expressed as:

$$\begin{aligned} \frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + R_y + \\ \mu_e \left[\frac{\partial}{\partial x} \left(\frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left(\frac{\partial v}{\partial z} \right) \right] + T_y + \rho f_y \end{aligned} \quad (4)$$

$$\begin{aligned} \frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + R_z + \\ \mu_e \left[\frac{\partial}{\partial x} \left(\frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left(\frac{\partial w}{\partial z} \right) \right] + T_z + \rho f_z \end{aligned} \quad (5)$$

Here R_x , R_y , R_z are distributed resistances which represent any source terms the user may wish to add. Distributed resistances are used to model the effect of some geometric feature without explicitly modeling the geometry, such as flow through porous media. The terms T_x , T_y , T_z are the viscous loss terms which are eliminated in the incompressible, constant property case. If the order of differentiation is reversed for each term it reduces to the derivative of the continuity equation, which is zero. The set of Eqs. (3-5) is called as Navier-Stokes equation.

The effective viscosity μ_e is the sum of laminar viscosity μ_l and turbulent viscosity μ_t *i.e.* $\mu_e = \mu_l + \mu_t$. For laminar flow $\mu_t = 0$, therefore effective viscosity is equal to the laminar viscosity or dynamic viscosity [8]. In ANSYS we can select the flow to be laminar or turbulent under solution option or by using FLDATA1 command. The value of laminar viscosity can be defined either in FLOTRAN setup using GUI or by FLDATA13 command [9]. The value of turbulent viscosity μ_t depends on turbulent model [8, 9] used in the fluid analysis. In the present work, standard k- ϵ Model is chosen for the analysis.

The energy equation is derived from the first law of thermodynamics. The energy equation in terms of stagnation temperature T_o for compressible flow can be written as:

$$\begin{aligned} \frac{\partial}{\partial t} (\rho C_p T_o) + \frac{\partial}{\partial x} (\rho u C_p T_o) + \frac{\partial}{\partial y} (\rho v C_p T_o) + \\ \frac{\partial}{\partial z} (\rho w C_p T_o) = \frac{\partial}{\partial x} \left(k \frac{\partial T_o}{\partial x} \right) + \frac{\partial}{\partial y} \left(k \frac{\partial T_o}{\partial y} \right) + \\ \frac{\partial}{\partial z} \left(k \frac{\partial T_o}{\partial z} \right) + W^v + Q_v + \Phi + \frac{\partial p}{\partial t} \end{aligned} \quad (6)$$

where Q_v , Φ , W^v and E^k are the volumetric heat source, viscous heat generation, viscous work and kinetic energy terms respectively. The expression for these terms is given below:

$$\Phi = \mu \left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i} \right) \frac{\partial u_i}{\partial x_k}, \quad T_o = T + \frac{V^2}{2C_p}$$

$$W^v = u_j \mu \left(\frac{\partial}{\partial x_i} \frac{\partial u_j}{\partial x_i} + \frac{\partial}{\partial x_k} \frac{\partial u_k}{\partial x_j} \right) \text{ and}$$

$$E^k = -\frac{\partial}{\partial x} \left[\frac{k}{C_p} \frac{\partial}{\partial x} \left(\frac{1}{2} |V^2| \right) \right] - \frac{\partial}{\partial y} \left[\frac{k}{C_p} \frac{\partial}{\partial y} \left(\frac{1}{2} |V^2| \right) \right] - \frac{\partial}{\partial z} \left[\frac{k}{C_p} \frac{\partial}{\partial z} \left(\frac{1}{2} |V^2| \right) \right]$$

2.1 Equation of Motion of an Airfoil

A typical cross section of an airfoil is shown in Figure 1. The airfoil properties correspond to per unit span which basically remain same at any arbitrary location of the section along the span. Here c is the chord length of the airfoil, h and α denote the heave (positive upward) and pitch (positive nose up) displacements respectively of point O . X_{cp} , X_{cm} and X_o are the positions of centre of pressure, centre of mass and centre of flexural axis respectively. The integrated aerodynamic forces are lift L (positive upwards) and pitching moment M (positive nose up) act at the flexural point O . The mass moment of inertia of the section about the flexure axis (point of support O) is given by I_α . The spring stiffness in heave and pitch motions are respectively denoted by K_h and K_α , while the corresponding structural damping coefficients are denoted by C_h and C_α .

For the airfoil section of mass m and moment of inertia about the center of mass I_{cm} per unit span, the kinetic energy T is given by [4]:

$$T = \frac{1}{2} m (\dot{h} - X_A \dot{\alpha})^2 + \frac{1}{2} I_{cm} \dot{\alpha}^2 \quad (7)$$

where $X_A = (X_{cm} - X_o)$

The potential energy V of the system is given by:

$$T = \frac{1}{2} K_h h^2 + \frac{1}{2} K_\alpha \alpha^2 \quad (8)$$

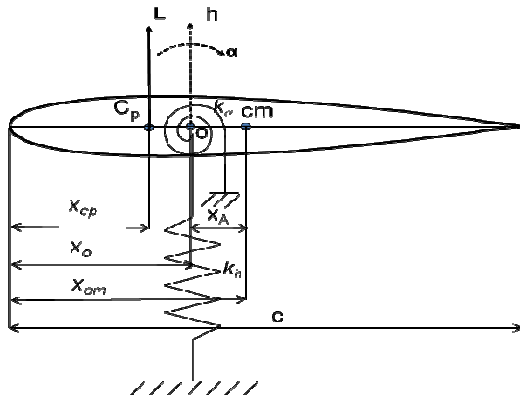


Figure 1. A NACA 0012 airfoil section

The equation of motion for the airfoil with two degrees of freedom (heave ' h ', and pitch ' α ') can now be obtained using the Lagrange's equation (for non conservative system) as:

$$\frac{d}{dt} \frac{\partial \bar{L}}{\partial \dot{u}_i} - \frac{\partial \bar{L}}{\partial u_i} + \frac{\partial (DE)}{\partial \dot{u}_i} = F_i \quad (9)$$

where $\bar{L} = T - V$ is the Lagrangian and DE is the dissipation energy of the system. By defining

$$DE = \frac{1}{2} C_h \dot{h}^2 + \frac{1}{2} C_\alpha \dot{\alpha}^2 \quad \text{we get the above equation}$$

in matrix form as [4]:

$$\begin{bmatrix} m & -mX_A \\ -mX_A & I_\alpha \end{bmatrix} \begin{Bmatrix} \ddot{h} \\ \ddot{\alpha} \end{Bmatrix} + \begin{bmatrix} C_h & 0 \\ 0 & C_\alpha \end{bmatrix} \begin{Bmatrix} \dot{h} \\ \dot{\alpha} \end{Bmatrix} + \begin{bmatrix} K_h & 0 \\ 0 & K_\alpha \end{bmatrix} \begin{Bmatrix} h \\ \alpha \end{Bmatrix} = \begin{Bmatrix} L \\ M \end{Bmatrix} \quad (10)$$

$$\text{or } [M]\{\ddot{u}\} + [D]\{\dot{u}\} + [K]\{u\} = \{F\} \quad (11)$$

where $[M]$, $[D]$ and $[K]$ are the mass damping and stiffness matrices of the system.

2.2 Time Integration Algorithm

Having obtained aerodynamic forces using the FEM based FLOTRAN CFD solver, Eq. (11) is solved numerically using Newmark's algorithm. Here, with discrete time steps, *i.e.* each of Δt , the displacement vector $\{u_{i+1}\}$ and the velocity vector $\{\dot{u}_{i+1}\}$ at time t_{i+1} are calculated.

$$\{u_{i+1}\} = \{u_i\} + \{\dot{u}_i\} \Delta t + \frac{\Delta t^2}{2} \{\ddot{u}_i\}$$

$$\{\dot{u}_{i+1}\} = \{\dot{u}_i\} + \frac{\Delta t}{2} [\{\ddot{u}_i\} + \{\ddot{u}_{i+1}\}] \quad (12)$$

The expression for the equation of motion at time t_{i+1} can be written as:

$$[M]\{\ddot{u}_{i+1}\} + [D]\{\dot{u}_{i+1}\} + [K]\{u_{i+1}\} = \{F_{i+1}\} \quad (13)$$

The above expression can be rearranged to get the acceleration at time t_{i+1} as:

$$\{\ddot{u}_{i+1}\} = \left[[M] + \frac{\Delta t}{2} [D] \right]^{-1} [\{F_{i+1}\} - [K]\{u_{i+1}\} - [D][\{\dot{u}_i\} + \frac{\Delta t}{2} \{\ddot{u}_i\}]] \quad (14)$$

The force vector $\{F_{i+1}\}$ at time $\{t_{i+1}\}$ is now obtained from FLOTRAN that simulates the aerodynamics for the given condition.

3. RESULTS AND DISCUSSION

First, the study of flow over NACA 0012 airfoil under steady flow condition is presented. These steady flow results are generated using ANSYS-FLOTTRAN under different flow conditions. The airfoil properties are listed in Table 1.

Figure 2 shows the finite element model of fluid domain over NACA 0012 airfoil. The boundary conditions applied over the airfoil surface and inlet boundary of the fluid domain are shown in Figure 3. The finite element mesh is formed using three noded linear triangular elements. The mesh near the boundary of airfoil is kept finer because of the presence of higher gradient of flow properties (p , ρ , T , V) and turbulence effects near the airfoil surface as compared to the outer boundaries of flow domain. The free stream boundary conditions applied at the upstream of the domain are u_∞ , v_∞ , ρ_∞ and p_∞ whereas zero pressure boundary condition is applied at the downstream of the domain. Different parameters defined in FLOTTRAN CFD solver for solving flow over an airfoil such as fluid properties, reference conditions and solution options are given in Table 2.

Table 1. Properties of the airfoil

Geometry	Profile: <i>NACA 0012</i> airfoil (symmetric).
Inertia Properties	$m=51.5 \text{ Kg}$, $I_\alpha = 2.275 \text{ Kg m}^2$ $X_0 = 0.4 \text{ m}$, $X_{cm} = 0.4429 \text{ m}$, $X_A = 0.0429 \text{ m}$ $I_{cm} = I_\alpha - mX_A^2 = 2.1802 \text{ kg m}^2$
Stiffness properties	$K_h = 50828.463 \text{ N/m}$, $K_\alpha = 35923.241 \text{ Nm/rad}$
Damping properties	$C_h = 32.358 \text{ Ns/m}$ (assuming $\zeta_h = \frac{C_h}{2\sqrt{K_h m}} = 0.01$) $C_\alpha = 5.718 \text{ Nms/rad}$ (assuming $\zeta_\alpha = \frac{C_\alpha}{2\sqrt{K_\alpha m}} = 0.01$)

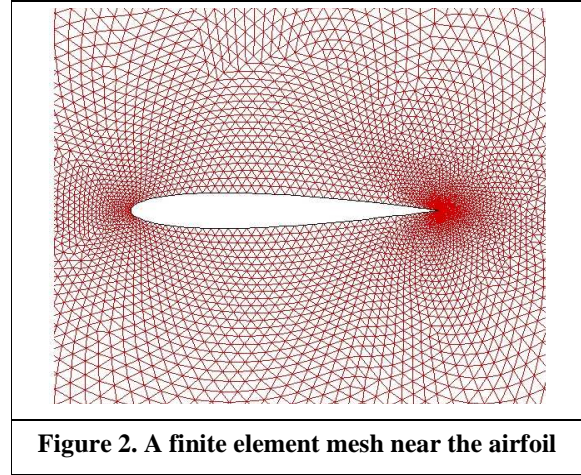


Figure 2. A finite element mesh near the airfoil

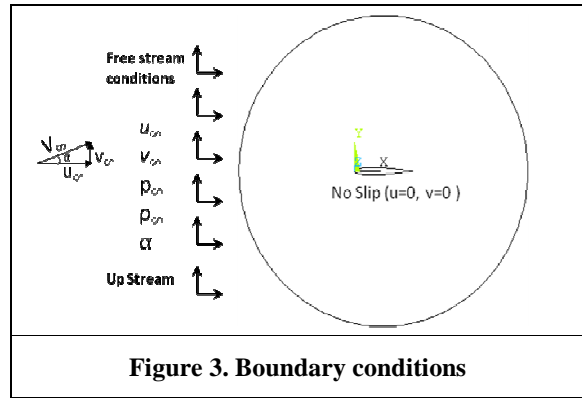
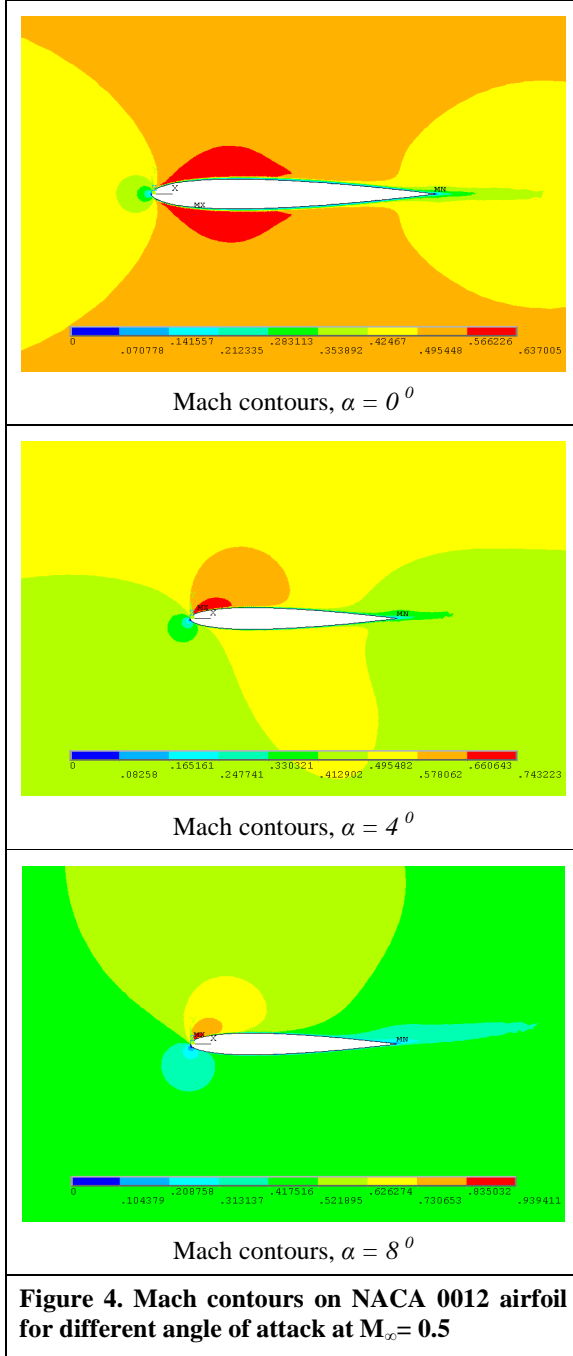


Figure 3. Boundary conditions

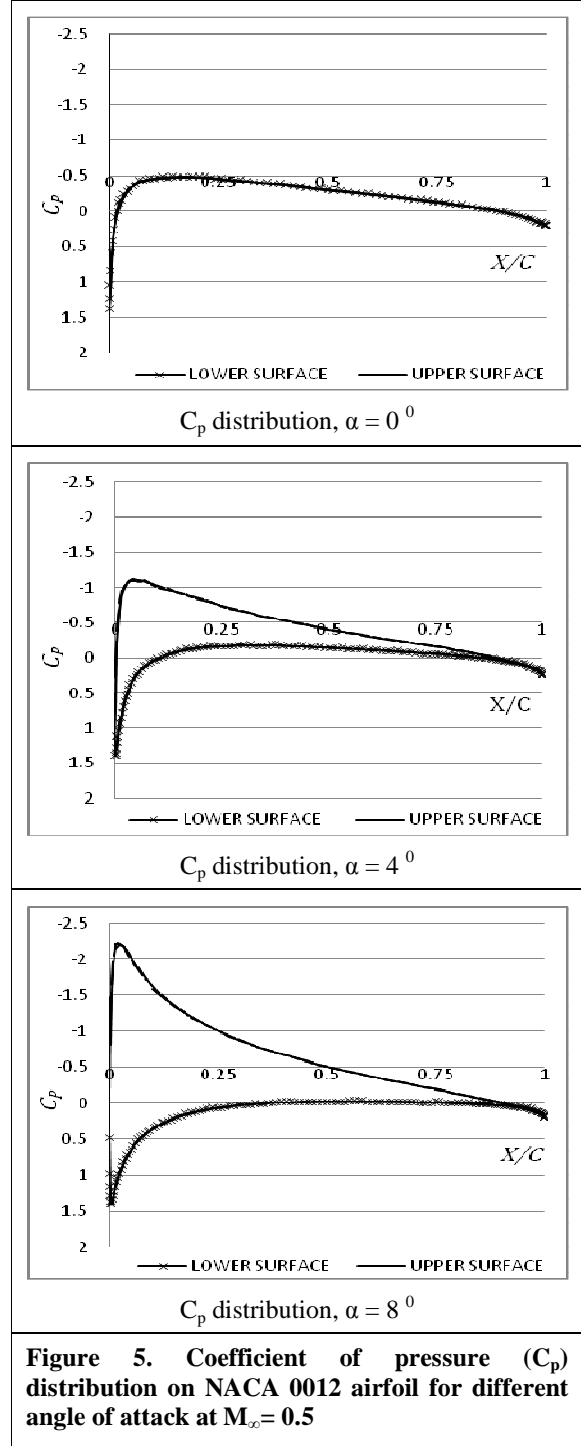
Table 2. Different parameters defined in FLOTTRAN for solving fluid flow over an airfoil

Fluid Properties	Density Viscosity Conductivity Specific heat	AIR in SI units
Reference conditions	Reference pressure Bulk modulus Ratio of C_p/C_v Nominal temperature Total stagnation temperature	101350 Pa $1\text{e}+015 \text{ N/m}^2$ 1.4 $= T + \frac{V^2}{2C_p}$ $= T + \frac{V^2}{2C_p}$
Solution options	Steady state or transient Adiabatic or thermal Laminar or turbulent Incompressible or compressible	Steady state Adiabatic Turbulent Compressible

The Mach contours and pressure distributions (C_p) over an airfoil at different angle of attack with $M_\infty = 0.5$ are presented in Figures 4 - 5 respectively. From the figures it is observed that at $\alpha = 0^\circ$ the net area enclosed by the curve of C_p against x/c is zero and hence there is no net aerodynamic force acting on the airfoil. However, at $\alpha = 4^\circ$ and $\alpha = 8^\circ$ the stagnation



point is on the lower surface of the airfoil and the area enclosed by the curve of C_p against x/c is not zero. The maximum value of pressure coefficient (C_p) increases with increase in angle of attack at any particular Mach number.



The variation of lift coefficient with angle of attack at constant free stream Mach number ($M_\infty = 0.44$) is shown in the Figure 6. From the figure, it can be seen that as the angle of attack increases, the lift coefficient increases linearly and the slope of C_L vs α curve remains constant. Since the effect of flow separation becomes dominant at higher angle of attack, the slope of the curve begins to fall off. Eventually the lift coefficient reaches a maximum value and then begins to decrease.

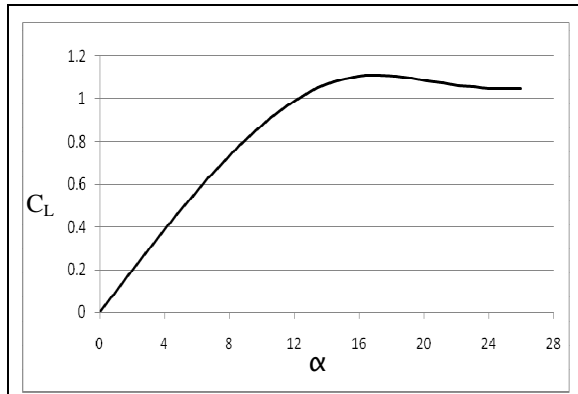


Figure 6. Variation of lift coefficient C_L with angle of attack for NACA 0012 airfoil

Next, the dynamics of NACA 0012 airfoil, with only structural damping is studied under different air flow conditions. The results are obtained by coupling ANSYS FLOTRAN with the in-house structural code in time domain using the Newmark's algorithm. For a given angle of attack, FLOTRAN simulates the pressure distribution over the airfoil which is later used to calculate the aerodynamic lift and moment.

These lift and moment are then passed to the structural code to estimate the response of the airfoil. Here, the value of structural damping ratios $\zeta_h = 0.01$ and $\zeta_\alpha = 0.01$ are taken into account. The time history of the dynamic response (heave and pitch) of the airfoil at the point of support and the corresponding aerodynamic lift coefficients for various free stream air flow velocities are presented in Figures 7 - 9.

From Figure 7, it is observed that the heave and pitch motions and the corresponding aerodynamic lift coefficient oscillate and converge to zero mean position of the airfoil with time at flow velocity of 190 m/sec. From Figure 8, it is evident that at the flow velocity of 192.45 m/sec, both the

heave and pitch motions of the airfoil are simple harmonic in nature and their amplitudes remain constant with time. This velocity is the flutter velocity of the airfoil system. However, from Figure 9, it can be seen that at a flow velocity of 193 m/sec, the airfoil oscillates unboundedly whose amplitudes increase exponentially with time. The results thus indicate that the oscillations of the airfoil under flow velocities below 192.45 m/sec is stable, and those beyond this critical velocity are unstable.

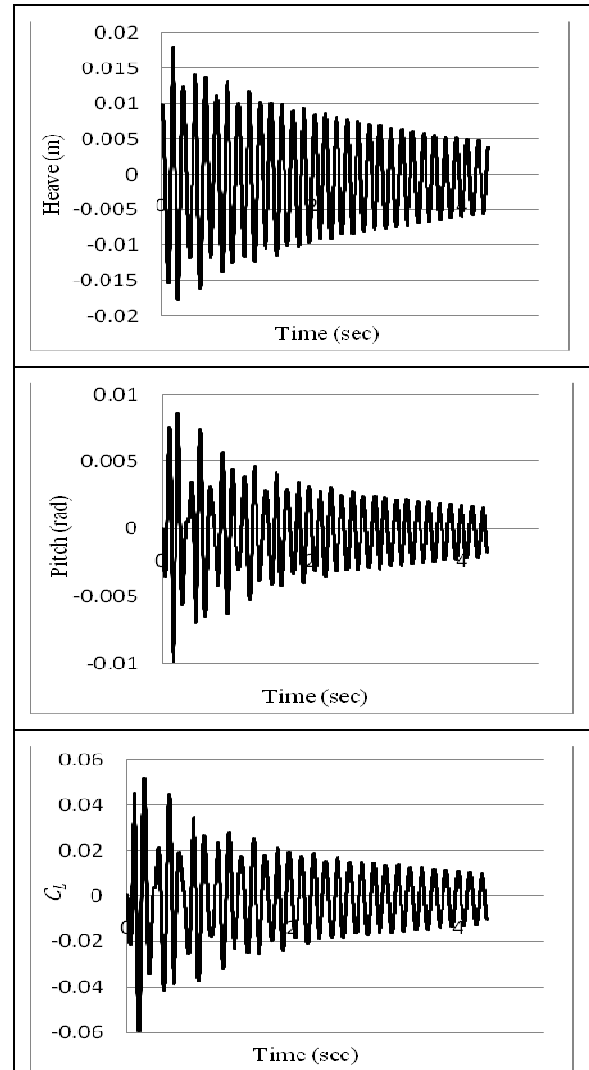


Figure 7. Time histories of heave and pitch motion and aerodynamic lift coefficient at the support point of airfoil with free stream velocity 190 m/sec

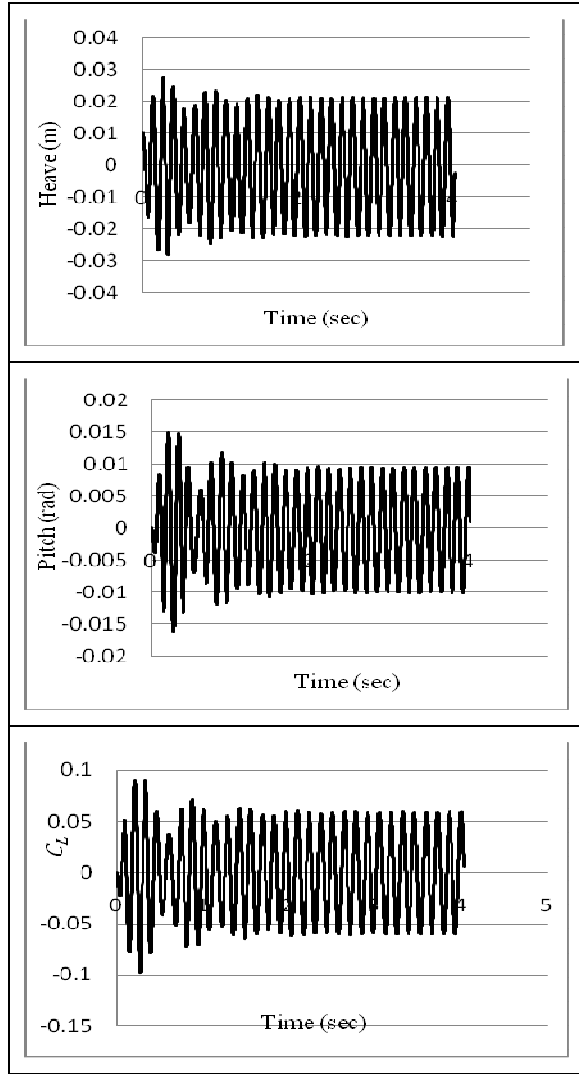


Figure 8. Time histories of heave and pitch motion and aerodynamic lift coefficient at the support point of airfoil with free stream velocity 192.45 m/sec

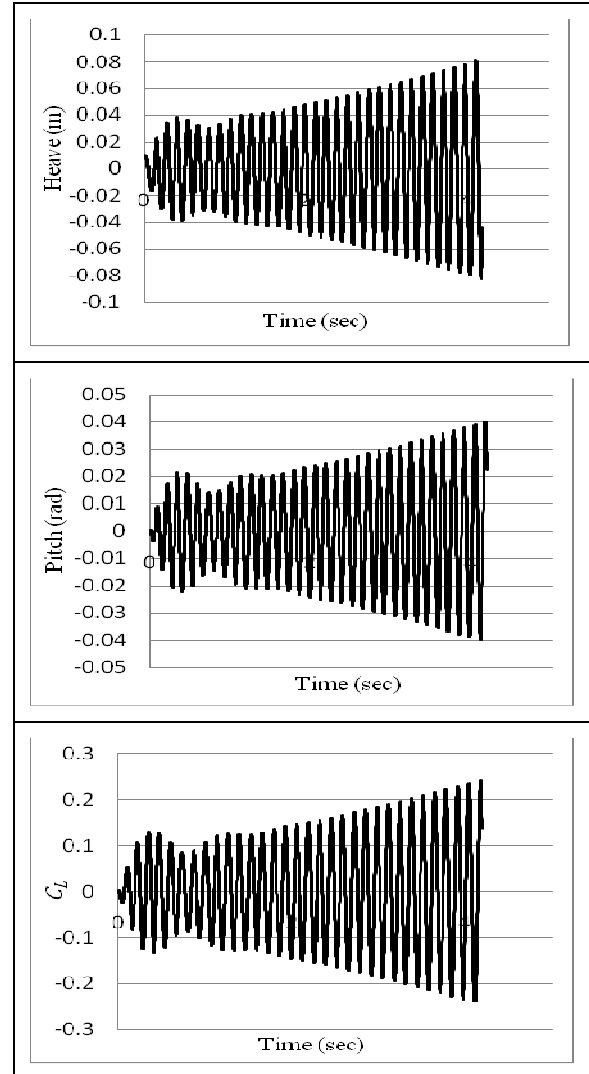


Figure 9. Time histories of heave and pitch motion and aerodynamic lift coefficient at the support point of airfoil with free stream velocity 193 m/sec

The flutter velocity obtained using the present coupled CSD-CFD code for NACA0012 airfoil has been also compared with those obtained using different time domain methods [4, 5]. From Table 3 it can be observed that the present flutter velocity is high compared to the velocities obtained using other time domain methods. This may be due to the fact that the present approach includes the turbulent viscous effect into flow. This leads to low prediction of aerodynamic forces for airfoil compared to results obtained in inviscid and laminar viscous flow.

4. CONCLUSION

In the present work, the dynamic response of an airfoil is studied by coupling ANSYS FLOTRAN CFD solver with the structural equations using direct integration method. Here, ANSYS FLOTRAN is adopted for solving the Reynolds Average Navier Stokes (RANS) equations based on k- ϵ turbulent model in quasi-steady flow condition. Then, a coupled CSD-CFD code is developed for studying dynamic behaviour of an airfoil in the time domain using Newmark's algorithm. In the present FSI study, the airfoil is not moved and the pitch angle of airfoil

is assigned to inlet flow velocity of the domain. Good agreement has been observed between the values of the flutter velocity obtained using present coupled CSD-CFD code and various other methods. The present flutter velocity is found to be high compared to the Euler and laminar viscous based CFD solver. This may be due to the fact that the present approach includes the turbulent viscous effect into flow which leads to low prediction of aerodynamic forces for airfoil compared to inviscid and laminar viscous flow. The present analysis with a simple airfoil in subsonic flow also indicates that it is possible to extend this work for predicting flutter even in the transonic and supersonic regimes. Furthermore, to model the complete unsteady effect in the flow, the actual airfoil motion has to be allowed and the mesh of fluid domain has to be moved accordingly.

Table 3. Flutter velocity of NACA0012 airfoil using different methods

Different Methods	Flutter Velocity
Time domain simulation using FEM based Euler solver (inviscid) for generation of aerodynamic forces in quasi steady flow [4]	174.20 m/s
Time domain simulation using FEM based N-S solver (viscous laminar) for generation of aerodynamic forces in quasi-steady flow [5]	184.55 m/s
Time domain simulation using ANSYS FLOTRAN CFD solver (viscous turbulent) for generation of aerodynamic forces in quasi steady flow	192.45 m/s

ACKNOWLEDGEMENT

The authors are grateful to the Head, STTD, NAL and the Director, NAL, for their generous support and encouragement for the work reported here.

REFERENCES

1. R. L. Bisplinghoff, H. Ashley. and R. L. Halfman, "Aeroelasticity", Addison-Wesley Publication 1957.
2. Y. C. Fung, "An Introduction to the Theory of Aeroelasticity", John Wiley and sons Inc, 1955.
3. D. H. Hodges, Pierce G. Alvin., "Introduction to Structural Dynamics and Aeroelasticity", Cambridge University Press, 2002.
4. M. Manjuprasad, Neeraj Kumar Sharma, Davinder Rana, and Amit Kumar Onkar, "Time Domain Simulation of Airfoil Flutter using Fluid Structure Coupling through FEM based Navier-Stokes Solver", NAL Technical Memorandum, TM-ST-09-03, November 2009.
5. Amit Kumar Onkar, A. Arun Kumar, M. Manjuprasad, "Flutter Prediction of an Airfoil using Fluid Structure Interaction in Time Domain through Finite Element Method based Navier-Stokes Solver", NAL Technical Memorandum, TM-ST-11 06, June 2011.
6. M. Manjuprasad, Amit Kumar Onkar, Davinder Rana, Neeraj Kumar Sharma, "Fluid Structure Coupling Algorithm for Prediction of Airfoil Flutter in Time Domain using Ansys-Flotran N-S Solver", TM-ST-10 02, July 2010.
7. J. D. Anderson, "Computational Fluid Dynamics: The Basics with Application", McGraw-Hill Book Company, First edition, 1988.
8. D. C. Wilcox, "Turbulence Modeling for CFD", DCW Industries Inc., La Canada, California, 1994.
9. ANSYS Tutorial, Complete Documentation - 2008.